CFD ANALYSIS OF ITER BLANKET MODULES

P. Asinari[1], S. Corpino[2], D. Marchisio[3], M. Merola[4], V. Rasetto[3], A. Rougier[2], L. Savoldi Richard[1], F. Subba[1], R. Zanino[1]

[1] *Dipartimento di Energetica, Politecnico di Torino, I-10129 Torino, Italy*
[2] *Dipartimento di Ingegneria Aeronautica e Spaziale, Politecnico di Torino, I-10129 Torino, Italy*
[3] *Dipartimento di Scienza dei Materiali e Ingegneria Chimica, Politecnico di Torino, I-10129 Torino, Italy*
[4] *ITER Organization, F-13067 Saint Paul lez Durance Cedex, France*

Corresponding author: roberto.zanino@polito.it

The International Thermonuclear Experimental Reactor (ITER) is under construction at Cadarache, France. The ITER blanket system consists of modular shielding elements known as Blanket Modules (BMs) which are attached to the Vacuum Vessel. Each BM consists of two major components: a plasma-facing First Wall (FW) panel and a Shield Block (SB) [1].

The BMs are actively cooled with pressurized water in forced circulation and require a careful thermal-hydraulic analysis as, e.g., FW panels are subject to heat loads up to 5 MW/m^2, implying that (highly) sub-cooled flow boiling conditions are expected, whereas SBs are cooled by single phase pressurized water. Nevertheless the complex cooling scheme of this component can hardly be computed analytically and requires the use of 3D hydraulic computational tools.

While other plasma-facing components, and in particular the divertor, have already been the object of a handful of papers devoted to their analysis using the tools of Computational Fluid Dynamics (CFD), see e.g. [2], [3], [4] very little work of this kind exists so far for the blanket.

In this paper we consider the two cases of the SB for BM #4 and the FW panel for BM #12. For the reference ITER operating scenario (resulting in a given heat load, pressure, inlet temperature, mass flow rate for these components) we apply the commercial ANSYS FLUENT code to the 3D hydraulic and thermal characterization of the two objects in terms of pressure drop and heat transfer coefficients, respectively. The detailed structure of the components is taken into account as defined in CATIA V5.

We concentrate in particular on the sensitivity to the choice of suitable physics models (e.g., for turbulence, multiphase where relevant, etc.) and on the comparison with existing correlations where available, after the verification of grid independence of the computed solution has been performed.

[1] M. Merola, presented at ISFNT-9, 2009.
[2] Y. Bournonville, et al., *Fusion Engineering and Design*, vol. 84**,** 2009, pp. 501–504
[3] S. Pascal-Ribot, et al., *Fusion Engineering and Design*, vol. 82**,** 2007, pp. 1781–1785
[4] A.R. Raffray, et al., *Fusion Engineering and Design*, vol. 45**,** 1999, pp. 377–407